Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 129 (2015) 308 – 316

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on Industrial Engineering

Simulation of electrical circuits using conjugate gradient algorithm

Burtsev Y.A.*

Platov South-Russian State Polytechnic University (NPI), 132, St. Prosvescheniya, Rostov region, Novocherkassk, 346428, Russian Federation

Abstract

Problems of electrical circuits' simulation are known for a long time. However, the simulation quality and speed are far from perfect, especially when it comes to non-linear circuits. The method offered is based on table form of electrical circuits' equations. Differential-algebraic equations of electrical circuits are converted to the systems of linear algebraic equations (SLAE) with finite differences method. It is important that SLAE are solved with Conjugate Gradient Algorithm (CGA) that is well adapted to systems with sparse matrixes. The solution of the SLAE at previous time step is a good initial approximation of the solution at present time step. That is why CGA reduces calculations to 20-40% of the full algorithm typically. The possibility of using CGA for solving SLAE with matrixes that have no specific sign is proved by numerical experiments. A method for acceleration of solving electrical circuits' SLAE is proposed. It differs from the Nodal Voltages Method as no apparent avatar of circuit SLAE is formed. A comparison of program "Electroscope" based on proposed method with programs "PSIM" and "Fastmean" is presented. "Electroscope" is leading in terms of quality and speed of test circuits' simulations.

Peer-review under responsibility of the organizing committee of the International Conference on Industrial Engineering (ICIE-2015)

Keywords: Electrical circuits; simulation; table form of equations; Conjugate Gradient Algorithm; matrixes with no specific sign; simulation speed; reliability; comparing.

1. Introduction

Problems of electrical circuits simulation and methods for their solving are known for long time. However, quality and speed of existing computer programs are far from perfect, especially for nonlinear circuits. For example, it is shown in article [1] that well known programs Micro-Cap 10.8.0, Multisim 10.0.144 and PSpice 9.1 produce solutions with rough defects, and by the way work much slower than programs Electroscope, Fastmean 5.2 and

* Corresponding author. Tel.: +7-908-505-30-39.
*E-mail address:*proton36@yandex.ru

Peer-review under responsibility of the organizing committee of the International Conference on Industrial Engineering (ICIE-2015)
doi:10.1016/j.proeng.2015.12.067

PSIM 9.0. That also can be said about program TINA Version 10.1.30.22 DT-DS build date 22 January 2015. As it is shown below, programs Fastmean 5.2 and PSIM 9.0 also generate solutions with defects and have not best simulation speeds. Program Micro-Cap does not allow contours of "voltage controlled elements" – ideal voltage sources or inductances, and their combinations. Programs Multisim, PSpice and Micro-Cap do not allow nodes that have no DC connection to grounded base node.

Existing programs for electrical circuit simulation cannot cover all needs of engineer calculations, because new devices often contain original components that are not provided by authors of programs. Circuits with switching and nonlinear elements and semiconductor devices which resistances vary by several orders are common in digital and power electronics. Nonlinear inductances of electrical machines that furthermore depend of the rotor rotation are also complicated circuit elements. Existing methods of circuit simulation are complicated and bulky, or they have considerable restrictions and are not universal. That is why development of new algorithms and programs that can effectively simulate electrical circuits for designing of new devices is an actual problem.

2. Simulation of Resistive Circuits

Popular practical electrical circuits' simulation methods usually are variants of the Nodal Voltages Method. However, for this method simulation of circuits with magnet coupling may have problems. Equation systems for circuits with big difference of resistances are poor conditioned.

So called table form of electrical circuits equations has some advantages [2, 3], main of them are simplicity and universality. Both voltages and potentials usually are present in the table form of equations. But voltages can be easily expressed via potentials and then they can be removed from the system. In this case an equation system of resistive circuit can be written in the form:

$$\begin{bmatrix} -\mathbf{R} & \mathbf{K}^{\mathrm{T}} \\ \mathbf{K} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{i} \\ \boldsymbol{\phi} \end{bmatrix} = \begin{bmatrix} \mathbf{e} \\ \mathbf{j} \end{bmatrix}, \tag{1}$$

where **R** is diagonal matrix of branch resistances; **K** is incidence matrix without string and column that correspond to basic node with zero potential; **0** is zero submatrix, **i** is vector of branch currents, **φ** is vector of node potentials; **e** is vector of EMF (electromotive force) sources; **j** is vector of current sources. Let us denote **A** matrix of the system (1).

For example, for the circuit shown at Fig. 1, the system (1) will have such appearance:

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & -R_1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -R_2 & 0 & 0 & 1 & 0 & 0 & -1 \\ 0 & 0 & 0 & 0 & -R_3 & 0 & 1 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -R_4 & 0 & 1 & 0 & 0 \\ 1 & 0 & 0 & 1 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & -1 & 1 & 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \\ i_5 \\ i_6 \\ \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_4 \end{bmatrix} = \begin{bmatrix} e_1 \\ e_2 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ j_1 \\ 0 \\ 0 \end{bmatrix}.$$

Matrix **A** is several times larger compared with the matrix of the Node Voltage Method, but it is very sparse and does not require any calculations for its forming. Its structure is so simple that it is no need to form it in any apparent avatar in the computer memory. It is sufficient only to know for each circuit element numbers of nodes to which it is connected.

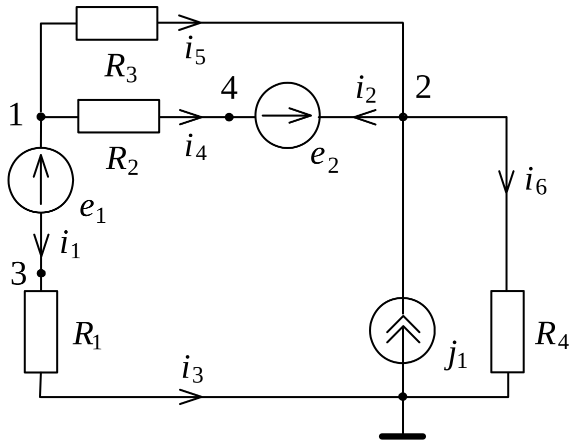

Fig. 1. Scheme of the electrical circuit.

It is principally important that structure of the matrix **A** is suitable for use Conjugate Gradient Algorithm (CGA) [4-12], that is very effective for solving of systems of linear algebraic equations (SLAE) with sparse matrixes. This algorithm does not change matrix of the SLAE, and thus it makes possible to use the sparsity completely. CGA uses multiple multiplication of vectors by matrix **A**, and also scalar multiplications of the vectors. Multiplication of vector by matrix **A** requires only N arithmetical multiplications and less than $4N$ additions, where N is number of circuit branches.

CGA is useful also by another reason. Voltages and currents of the electrical circuit are continuous or piecewise-continuous functions. That is why state of the circuit at previous time step usually is a good initial approximation for its state at present time step, and CGA requires little number of iterations – typically 20-40% of the matrix **A** dimension. In addition, CGA can be easy combined with parallel calculations.

Researches [13] have shown that probability of CGA fail during solving SLAE with symmetrical matrix with no specific sign is vanishingly small, though it is not zero. Practically, only special constructing of particular case can lead CGA to fail due to critically little value of denominator $(\mathbf{A}\mathbf{s}_k, \mathbf{s}_k)$ in one of the CGA formulas, here $\mathbf{s}_k$ is "direction vector" of the CGA. But even if CGA fails, the fail can be easily and quickly detected in the computer program, for its correction it is sufficient to take another initial approximation.

Singularity of the matrix **A** also does not prevent using CGA. This fact is proven by simulations of electrical circuits in which submatrix **K** was full incidence matrix of the circuit graph (including all nodes), and also by simulations of the circuits that were sectioned into insulated parts (with "floating" potentials). Circuits consisting of separate parts, present one of two variants of topological degeneration. It is important that circuits at the edge of such degeneration often appear in practical computations. For example, a diode rectifier working in parallel with load and capacitor, periodically closes all of the diodes and thus separates source from the load.

Let us describe using of proposed method for simulation nonlinear circuits. Let us suppose that we have some initial approximation of branch currents and nodal potentials of the circuit. Using these values, we can calculate static resistances of nonlinear branches and insert them into the system (1). Having solved it, we get the next approximation of the currents and potentials. Then we shall continue the process until desired accuracy will be reached. This practical method of nonlinear circuit calculation is known in electrical engineering and is briefly presented, for example, in the book [14]. This method is called in mathematical literature as "method of false position", "method of linear interpolation", or "regula falsi". All currents and potentials are known at each time step. It makes possible easy using of volt-ampere characteristics of the elements either if they are presented as voltages depending on current, or as currents depending on the voltages.

This method always converged after few number of iterations, usually 1, 2 or 3, for huge number of simulations of different circuits with diodes, transistors and nonlinear inductances.

3. Simulation of Circuits with Reactive Elements

Equation systems of the electrical circuits with reactive elements are systems of differential-algebraic equations (DAE). Finite-difference method [6] makes possible to convert DAE systems into SLAE. Three-point backward differentiation formula (BDF) generally gives solutions with quite satisfactory accuracy if we do not consider such cases as long transient processes in the circuits with high reactance factor. For example, tree-point BDF of the inductance current calculated at the time moment t looks like this:

$$i'_L(t) \approx \frac{i_L(t-2\Delta t) - 4i_L(t-\Delta t) + 3i_L(t)}{2\Delta t}\,, \tag{2}$$

where i_L is current of the inductance, t is time, Δt is time step.

Derivation of the capacitor voltage can be obtained like formula (2). Taking into account that

$$u_C(t) = \phi_{C1}(t) - \phi_{C2}(t)\,,$$

where u_C is voltage of the capacitor, φ_{C1} and φ_{C2} are potentials of the nodes to which the capacitor is connected, we gain:

$$\begin{aligned} u'_C(t) &\approx \frac{u_C(t-2\Delta t) - 4u_C(t-\Delta t) + 3u_C(t)}{2\Delta t} = \\ &= \frac{\phi_{C1}(t-2\Delta t) - \phi_{C2}(t-2\Delta t) - 4(\phi_{C1}(t-\Delta t) - \phi_{C2}(t-\Delta t)) + 3(\phi_{C1}(t) - \phi_{C2}(t))}{2\Delta t}\,. \end{aligned} \tag{3}$$

After substitution of expressions (2) and (3) into equations of inductance and capacitor respectively, we gain equations similar to the equations of resistive branches:

$$-\frac{3L}{2\Delta t} i_L(t) + \phi_{L1}(t) - \phi_{L2}(t) = \frac{L}{2\Delta t}\left[i_L(t-2\Delta t) - 4i_L(t-\Delta t)\right]\,, \tag{4}$$

$$\begin{aligned} &-\frac{2\Delta t}{3C} i_C(t) + \phi_{C1}(t) - \phi_{C2}(t) = \\ &= -\frac{1}{3}\left[\phi_{C1}(t-2\Delta t) - \phi_{C2}(t-2\Delta t)\right] + \frac{4}{3}\left[\phi_{C1}(t-\Delta t) - \phi_{C2}(t-\Delta t)\right], \end{aligned} \tag{5}$$

where L is inductance, C is capacitance, i_C is current of the capacitor, φ_{L1}, φ_{L2} are potentials of the nodes to which the inductance is connected.

Equations (4, 5) make possible to write an equation system of electrical circuit with reactive elements in form (1) for an equivalent resistive circuit, and then solve it by the CGA at each time step. The coefficients of the currents in the equations (4, 5) play role of quasi-resistances. Adding magnet couplings to the circuit inductances only makes submatrix **R** non diagonal, but does not change something essentially.

In equations of the circuits with time-dependent (including nonlinear) reactive elements there is necessary to consider capacitances and inductances as differential parameters, calculated in some point of the charge-voltage or flux-current characteristic:

$$C = \frac{dq}{dt}, L = \frac{d\Psi}{dt}.$$

In this case there are no derivations of the parameters C and L present in the differential equations of the capacitors and inductances, the equations preserve their common form:

$$i = \frac{dq}{dt} = C\frac{du}{dt}, \quad u = \frac{d\Psi}{dt} = L\frac{di}{dt}.$$

If capacitance and inductance are considered as static parameters

$$C = \frac{q}{u}, \quad L = \frac{\Psi}{i},$$

then their derivations also must be taken into account in the equations. That makes equations too complicated with no need.

4. Acceleration of Simulation by Substitutions

If matrix **R** is diagonal and does not contain little resistances, it can be easy inverted. In this case multiplication by the matrix **A** that is necessary in CGA, can be done in two steps. First, we calculate fictive currents that are caused by branch voltages without sources:

$$\mathbf{i}_0 = \mathbf{R}^{-1}\mathbf{K}^{\mathrm{T}}\phi.$$

Then we substitute these currents into equations of the first Kirchhoff law and find vector

$$\phi_0 = \mathbf{K}\mathbf{R}^{-1}\mathbf{K}^{\mathrm{T}}\phi.$$

Right part of the system (1) must be previously conversed to the form

$$\begin{bmatrix}\mathbf{0} & \mathbf{j} + \mathbf{K}\mathbf{R}^{-1}\mathbf{e}\end{bmatrix}^{\mathrm{T}}.$$

That correspond converting EMF sources to the current sources and temporary replacing real currents **i** with fictive currents $\mathbf{i}_0$. As result of all these transformations, only vector of potentials φ will be found as solution of the SALE, and thus number of unknown variables and number of the SLAE equations will be reduced comparing to immediate solving the system (1). Real branch currents are calculated after solving SLAE as

$$\mathbf{i} = \mathbf{R}^{-1}(\mathbf{K}^{\mathrm{T}}\phi + \mathbf{e}).$$

Described procedure gives a method of multiplication of vector φ by matrix of the Node Voltages Method equation system that looks like

$$\mathbf{K}\mathbf{R}^{-1}\mathbf{K}^{\mathrm{T}}\phi = \mathbf{j} + \mathbf{K}\mathbf{R}^{-1}\mathbf{e}.$$

Such multiplication is useful for solving this system by the CGA. Difference between proposed procedure and the Node Voltages Method is not calculating of the SLAE matrix in any apparent form. That considerably reduces calculations and simulation time. A variant of such substitutions for circuits with very little and zero resistances, and also with magnet coupling is developed and successfully tested.

Besides that, simulation time can be reduced by eliminating of branches that consist only of ideal EMF sources. It can be done by transferring of the sources via the nodes (reduction of the EMF sources).

5. Comparing of the Programs

Fig. 2 and Fig 3 show test circuits for comparing program "Electroscope" based on proposed method, with two most qualitative and fast programs of considered in article [1] – with American program PSIM Professional Version 9.0.3.400 and Russian program Fastmean ver. 5.2 (issued Oct 30 2012/08:38:55). EMF sources of first test circuit give square pulse voltage with minimum -10 V and maximum 10 V, duty cycle 50%, frequencies are 50 Hz and 111 Hz. All EMF sources of the second test circuit give square pulse voltage with minimum -400 V and maximum 400 V, duty cycle 50%, frequencies are10,1; 10,3; 10,7; 10,9; 11,3; 12,7 and 13,1 kHz.

Program "Electroscope" has only nonlinear diode models. Programs PSIM and Fastmean can use either simple linear diode models or nonlinear models. Tables 1 and 2 show simulation time for different time steps (all of the programs use fixed time step).

There are used such notifications for variants of the "Electroscope" program: "Cur+EMF" – for program with acceleration by substitution of fictive currents and EMF reduction; "Cur" - for program with acceleration by substitution of fictive currents without EMF reduction; "Sys (1)" – program with immediate solving of the SLAE (1).

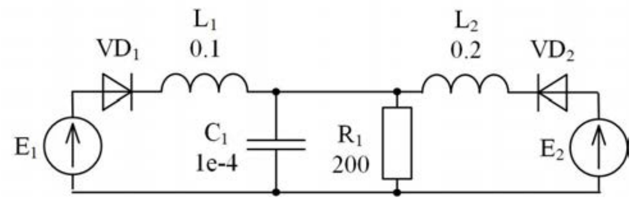

Fig. 2. Test circuit # 1.

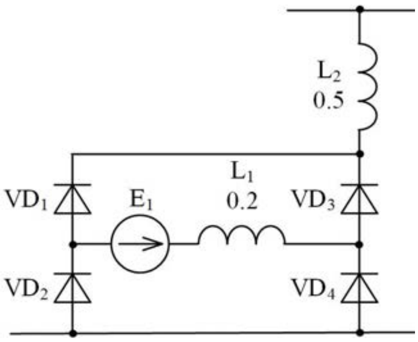

Fig. 3. A part of the test circuit # 2. The whole circuit consists of 7 such parts connected in parallel to resistor 10 Ohm.

Table 1. Comparing of simulation time (in seconds) for test circuits. Modeling time 0.2 s, time step 1 us.

	Electroscope			Fastmean		PSIM	
Circuit #	Cur + EMF	Cur	Sys (1)	Linear model	Nonlin. model	Linear model	Nonlin. model
1	1,031	1,531	5,484	1,750	3,312	2,3	57
2	5,719	12,391	31,265	12,89	13,94	10,3	111

Table 2. Comparing of simulation time (in seconds) for test circuits. Modeling time 2 s, time step 10 us.

	Electroscope			Fastmean		PSIM	
Circuit #	Cur + EMF	Cur	Sys (1)	Linear model	Nonlin. model	Linear model	Nonlin. model

1	1,141	1,547	6,344	1,828	3,094	2,0	50
2	13,297	16,734	93,125	34,36	33,218	17,8	112

It is critically important that program "Electroscope" produces solutions without rough defects. Programs PSIM and Fastmean show significant (double and more) false peak overvoltages of the diodes. Program PSIM apparently uses Newton-Raphson method for solving nonlinear equations. During simulation of both test circuits it gives numerous warnings "The program fails to converge after 10 iterations when determining switch positions". Quality of simulation results you can evaluate from Fig. 4 – Fig. 6.

Fig. 4. Program "Electroscope". Circuit # 1, voltage of diode VD_2 [V] vs time [ms], time step 1 us.

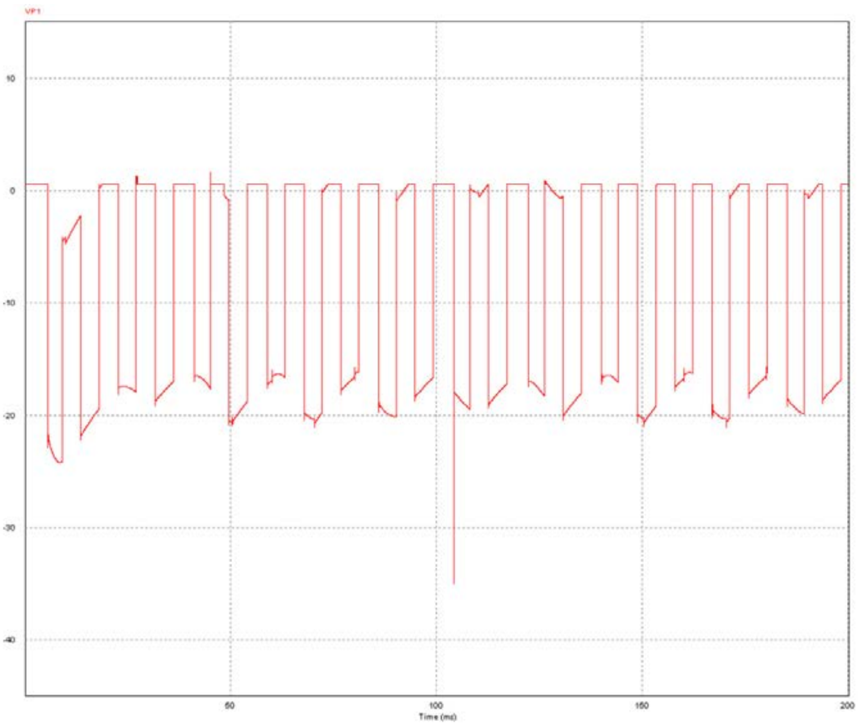

Fig. 5. Program PSIM. Circuit # 1, voltage of diode VD_2 [V] vs time [s], time step 1 us. Linear model of the diodes.

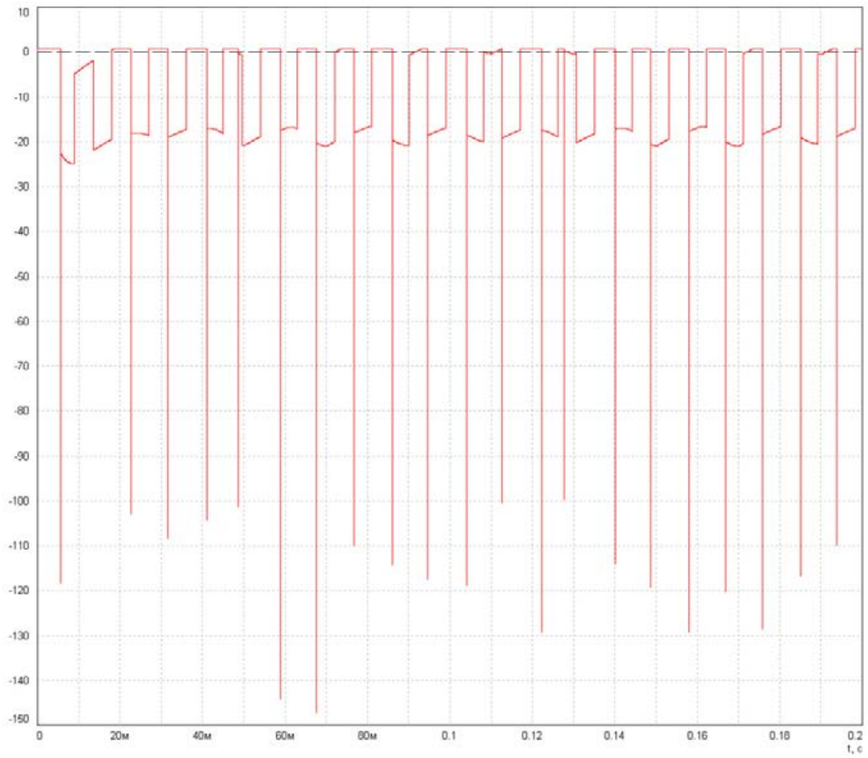

Fig. 6. Program Fastmean. Circuit # 1, voltage of diode VD_2 [V] vs time [s], time step 1 us. Linear model of the diodes.

6. Summary

Proposed method of electrical circuit simulation is universal and makes possible simulation of any processes in any electrical circuits consisting of two-terminals. It makes simulation simply, fast and with little computational work. Resistances of the two-terminals can vary by some orders, that situation is typical for circuits with nonlinear and switching elements. Circuits divided into several insulated parts or circuits at the edge of such condition also can be simulated. Multi-terminal elements of the electrical circuits can be presented as equivalent connections of two-terminals.

Possibility of simulation of electrical circuits with arbitrary magnet couplings is one of important advantages of the proposed method comparing with widely used Nodal Voltages Method. Apparently, limitations of the last are the main reason of absence possibility of arbitrary magnet coupling in many electrical circuit simulation programs. Meanwhile, presence of complicated magnet couplings is typical for example, for transformers and electrical machines.

Proposed method has high simulation speed and reliability that can be seen from comparing of program "Electroscope" with other programs. Defects in their simulation are caused probably by incorrect using of finite-difference formulas, by lack of accuracy in calculations of the time moments of diode switching, and also by failings of convergence in processes of solving the nonlinear equations.

Let us notice that developed mathematical apparatus may be useful not only for electrical circuits simulation, but also with help of the well known analogies it can be used for simulation of hydraulical, pneumatical and mechanical systems, and also for simulation of heat spreading in solid state constructions. To this purpose the corresponding objects must be presented as equivalent electrical circuits. In addition, there are methods for converting various problems of physical field computations into problems of electrical circuits' computations that can be solved by proposed method.

References

[1] Y.A. Burtsev. Comparing of program for simulation of electrical circuits at the base of modified table method with known analogs, Izvestiya vyshih uchebnyh zavedenij. Elektromehanika. 4 (2013) 8─13.
[2] A.I. Petrenko, A.I. Vlasov, A.P. Timchenko, Table Methods of Electronic Schemes Computer Modelling, Visha shkola, Kiev, 1977.
[3] J. Vlach, K. Singhal, Computer Methods for Circuit Analysis and Design, Radio i Svyaz, Moscow, 1988.
[4] L. Hageman, D. Yong, Applied Iterative Methods, Mir, Moscow, 1986.
[5] N.S. Bakhvalov, N.P. Zhidkov, G.M. Kobelkov, Numerical Methods, Science, Moscow, 1987.
[6] G. Korn, T. Korn, Mathematical Handbook for Scientists and Engineers, fourth ed., Science", Moscow, 1978.
[7] I.M. Vinogradov, Mathematical Encyclopedy, vol. 5, Sovetskaya Encyclopediya, Moscow, 1984.
[8] G.I. Marchuk, Methods of Calculus Mathematics, third ed., Nauka, Moscow, 1989.
[9] I.S. Berezin, N.P. Zhidkov, Methods of calculation, State Publishing of Physical and Mathematical Literature, Moscow, 1962.
[10] D.K. Faddeyev, V.N. Faddeyeva, Calculus Methods of Linear Algebra, State Publishing of Physical and Mathematical Literature, Moscow, Leningrad, 1963.
[11] V.V. Voevodin, Numerical Methods of Algebra (theory and algorithms), Nauka, Moscow, 1966.
[12] R. Hockney, J. Eastwood, Computer Simulation Using Particles, Mir, Moscow, 1987.
[13] Y.A. Burtsev, Numerical-experimental Research of Use Conjugate Gradient Algorithm for Solving Electrical Circuit Equations in Table Form, Izvestiya vyshih uchebnyh zavedenij. Elektromehanika. 3 (2013) 8─15.
[14] A.E. Kaplyansky, Fundamental Theory of Electrical Engineering, second ed., Vysshaya shkola, Moscow, 1972.